RESPONSE OF HUMAN BONE SYNTHETIC UNDER IMPACT LOAD USING FINITE ELEMENT METHOD

MOHAMAD SHAHNAZ BIN SAMSUL BAHARI

Thesis submitted in fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature
Name of Supervisor: Dr Daw Thet Thet Mon
Position: Supervisor
Date: November 2009

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature:
Name: Mohamad Shahnaz Bin Samsul Bahari
ID Number: MA06050
Date: November 2009

COPYRIGHT

I humbly dedicated this thesis to

my lovely father and late mother, Samsul Bahari Bin Abd Razal and Zarah Binti Sulaiman

and my siblings

who always trust me, love me and had been a great source of support and motivation.

ACKNOWLEDGEMENTS

This thesis is the end of my long journey in obtaining my Bachelor Degree in Mechanical Engineering. I have been accompanied and supported by many people in this journey. It is a pleasant aspect that I have now the opportunity to express my gratitude for all of them.

First a very special thanks you to my supervisor, Dr. Daw Thet Thet Mon for her great patient and efforts to explain things clearly. She provided encouragements and lots of good ideas in order for me to complete the analysis and thesis writing period. I would have been lost without her guidance and support.

I would also indebted to the support of my family and friends. They accompanied me went through the toughest moment. A journey is easier when you travel together. Interdependence is certainly more valuable than independences.

ABSTRACT

Commercial bone synthetic can be considered as a composite material where testing can be carried out to find out its mechanical properties. Generally there is lack of information about the properties of bone synthetic although it has been extensively used in caring human bone defect and injuries. Particularly, its real strength to external load, particularly sudden load is unknown. This project focus on identify the response of human bone synthetic under impact loading using finite element method. The geometry of finite element model was constructed in SolidWorks mechanical CAD package. The micron size of pores was created spline-feature-extrude method to include porosity as close as possible to that of the actual bone. The computational model was analyzed using ALGOR. The appropriate model was determined through the mesh convergence analysis for better accuracy. The results were is good agreement in term of location of their concentration. The proposed finite element model can be used to analyze the response of the human bone under various loading.

ABSTRAK

Komersial tulang sintetik boleh dianggap sebagai material komposit di mana pengujian boleh dilakukan untuk mengetahui sifat mekanik nya. Secara umumnya terdapat kurangnya maklumat tentang sifat-sifat tulang sintetik meskipun telah banyak digunakan dalam merawat tulang manusia cacat dan luka-luka. Terutama, kekuatan sebenarnya beban luaran, terutama beban yang tidak diketahui. Projek ini tertumpu pada respon mengenalpasti tulang sintetik manusia di bawah pengaruh beban menggunakan *'finite element method'*. Geometri model dibuat dalam *'Solidwork CAD'* pakej. The mikron saiz porositi dibuat menggunakan kaedah '*spline-feature-extrude'* untuk menjadikan porositi sehampir mungkin dengan tulang yang sebenarnya. Model yang digunakan dianalisis dengan menggunakan *'ALGOR'*. Model yang sesuai ditentukan melalui analisis '*mesh convergence'* untuk mendapat akurasi yang lebih baik. Keputusan adalah baik dimana beban-beban adalah tertumpu sama seperti keputusan experiment. Model '*finite element'* boleh digunakan untuk menganalisis respon daripada tulang manusia dalam pelbagai beban.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAD	Computer Aided Design
FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Model
FYP	Final Year Project
MES	Mechanical Event Simulation

LIST OF NOMENCLATURE

E	Modulus of Elasticity
σ	Stress
ε	Strain
L	Length
D	Diameter
α	Ducker-Prager Yield Function Alpha
β	Ducker-Prager Yield Function Beta
C	Damping Coefficient
δy	Yield Stress
F	Force
N	Newton

CHAPTER 1

INTRODUCTION

1.1 OVERVIEW

Human bone can be considered as a composite material where testing can be carried out to find out its mechanical properties. Generally there is lack of information about the properties of bone. This situation arises because of few studies conducted on the human bone as it is not easy to get the specimen. Nowadays, human bone can be replaced by bone synthetic. Bone synthetic is claimed as successor to the real bone. The synthetic bones also are made from composite material. The importance application of bone synthetic is to repair and healing at the site of a bone defect. It is successfully used in medical field because synthetic bone is no risk of immune response and no risk of disease transmission.

Few researchers have used Finite Element Analysis (FEA) to find out properties of natural bone. Finite Element Analysis (FEA) plays a bigger role in the analysis and testing of material virtually using FEA software such as ALGOR, COSMOS and LS-DYNA. The benefits of using FEA are modeling of complex boundary condition and material models and realistic simulation of variety of problems. Besides that, the advantage of using FEA is that it is economical by reducing the number of experiment FEA allows more intricate analysis of material besides making it easy to conduct different tests to a material [Hutton, David V.,2004]. In this project FEA is used to find out the response of human bone synthetic under impact load.

1.2 PROBLEM STATEMENT

Commercial bone synthetic has been extensively used in caring human bone defect and injuries. However, its real strength to external load, particularly sudden load is unknown. Extensive experimental work is too expensive.

1.3 OBJECTIVE

The objectives of this project are:

i. To develop a computational model of bone synthetic using Finite Element Method (FEM).
ii. To simulate the response of bone synthetic under impact load.
iii. To establish response curve for bone synthetic subjected to impact load.

1.4 PROJECT SCOPE

The scope of the project is:

i. Model geometry will be created in Solidwork
ii. Computational model will be developed using ALGOR.
iii. Computational model will be validated with experiment result done by others.
iv. Response of bone synthetic to impact load will be analyzed.
v. Response curve will be developed from simulated results.

1.5 ORGANISATION OF THIS THESIS

This thesis consists of five chapters. Chapter one is highlights background of the project, problem statement, objectives and project scope. Chapter two is literature reviews which study about researchers has done before this and make it as references and guide line for this project. Chapter three is for the methodology which is the explanation for the flow and step of this project. Chapter four is for the result and discussion. In this chapter is discussing the result for the analysis and validated the result. The last chapter is chapter five which is conclusion and recommendation for this project.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

As a part of the project, the analysis of literature is important as to have a further understanding the project. The materials that used for the literature review are from the internet web pages, journals, and from the books. The review is to find out the relevance of the project and it must have a significant relation with the project.

2.2 HUMAN BONE

Bone is a hybrid of organic and inorganic composite containing protein and mineral with superior hardness, strength and fracture toughness [Huajian Gao,Baohua Ji, 2003]. It is a rigid organ which is a part of human endoskeleton, it functions to enable movement, provide support and also to protect vital organs as it has a very hard feature. Besides that, human bones also function to store minerals, produces red and white blood cells [Enderle, John D., Bronzino, Joseph D.,and Blanchard, Susan M., 2005]. Bones has many shapes and has a very complex structure both internal and external. Bone tissue or osseous tissue makes up as the element of bone structure which gives its rigidity and honeycomb internal structure. [Wikipedia/Bone].

2.2.1 Structure and Characteristic of Human Bone

The bone is made up of calcium phosphate as its main structural material. Human bone is generally classified into two types that are: cortical (compact) bone and Trabecular or Cancellous bone [www.engin.umich.edu/class/bme456/bonestructure]. It is classified based on its porosity level and microstructure. Generally the compact bone or cortical bone has a very denser structure and its porosity level is ranging from 5% to

10%. From overall human body's bone mass, cortical makes about Cortical bone is very compact and has a very high strength. Thus it forms the outer layer around the Trabecular bone in shaft of long bones. Trabecular or cortical bone is much porous with porosity level in between 50% to 90%.It accounts for the remaining 20% of the total bone mass and has almost ten times of the surface area of compact bone. [Wikipedia/bone].

The cortical bone as other biological tissues has a hierarchical structure that is bone contains many different structures. On general it has been classified into three main type based on its microstructure organization.

Bone has up to seven hierarchical level of organization from the microstructure of the mineral crystals to the microstructure of cancellous and cortical bone [Huajian Gao, Baohua Ji, 2003].This mean the cortical bone has different structure layers similar to a Russian dolls. [Enderle, John D.,Bronzino, Joseph D., and Blanchard , Susan M., 2005].

The average trabecular Young's modulus measured ultrasonically and mechanically was 14.8 GPa (S.D. 1.4) and 10.4 (S.D. 3.5) and the average Young's modulus of microspecimens of cortical bone measured ultrasonically and mechanically was 20.7 GPa (S.D. 1.9) and 18.6 GPa (S.D. 3.5). With either testing technique the mean trabecular Young's modulus was found to be significantly less than that of cortical bone ($p < 0.0001$). [Rho JY, Ashman RB, Turner CH, J biomech, 1993]

Figure 2.1: Human Bone Structure

2.3 BONE SYNTHETIC

Bone synthetic is a piece of bone used to take the place of a removed bone or bony defect. Bone grafting is a surgical procedure that replaces missing bone with material from the patient's own body, an artificial, synthetic, or natural substitute. Bone grafting is used to repair bone fractures that are extremely complex, pose a significant health risk to the patient, or fail to heal properly. [Wikipedia/Bone_grafting]

Every method of bone grafting has its own advantages and disadvantages (pain, scarring, infection, and cost), but there are many viable options and patients are well-served to be educated about their choices regarding this aspect of a spinal fusion, and proactively work together with their surgeon to select the option that is right for them [Ebraheim NA, Elgafy H, Xu R, 2001]

2.3.1 Bone Synthetic Materials

Artificial bone can be created from ceramics such as calcium phosphates (hydroxyapatite and tricalcium phosphate), Bioglass and calcium sulphate; all of which are biologically active to different degrees depending on solubility in the physiological environment[Wikipedia/Bone_grafting]. These materials can be doped with growth factors, ions such as strontium or mixed with bone marrow aspirate to increase biological activity. Some authors believe this method is inferior to autogenous bone grafting [Jeffery C, Wang, MD, Bone Graft: New Development, 2009].

Artificial synthetic bones are made of HTR polymer, which fosters the growth of new bone. Synthetic bone grafts are implemented in several sectors: if bone and periodontium defects are to be corrected, when the maintenance of bone tissue needs to be fostered, and in situations where the dental lamina needs to be raised [www.alfieridentista.it/eng/synthetic-bones-grafts.html].

2.3.2 Autogenous Bone Graft

The gold standard of bone grafting is taking the patient's own bone. This is called autogenous bone graft. This means that at the time of surgery, the surgeon makes a separate incision and takes a small piece of bone from an area of the body where it is not needed. Typically, autogenous bone grafts are taken from the pelvis or iliac crest. Autogenous bone grafting has excellent fusion rates and has become the standard by which all other biologics are measured. Many surgeons prefer autogenous bone grafts because there is no risk of the body rejecting the graft since it came from the patient's own body [Jeffery C, Wang, MD, 2009].

2.3.3 Allograft Bone Graft

In an effort to minimize the problems associated with taking the patient's own bone, a number of other fusion techniques have been developed that use biological products as bone graft extenders or as bone graft replacements. One common source of bone graft replacement or extender is the use of allograft bone. An allograft bone graft is bone harvested from cadavers or deceased individuals who have donated their bone for use in the treatment of living patients. This is commonly used in many forms for

spinal fusions ranging from cervical interbody fusions to lumbar interbody fusions and can provide excellent structural support [Jeffery C, Wang, MD, 2009].

2.4 IMPACT LOADING

Impact testing is testing an object's ability to resist high-rate loading. An impact test is a test for determining the energy absorbed in fracturing a test piece at high velocity.

Most ductile materials have strength properties which are a function of the loading speed. The more rapid loading, give the higher tensile and ultimate strengths of the materials. Two standard tests of the impact loading are the Charpy and Izod. This two test can measure the impact energy which is the energy required to fracture a test piece under an impact load. It also called the notch toughness.

The detailed assessment of the strength of machine elements under impact loading regimes involves use of advanced techniques including Finite Element Analysis. Impact loads result in shock waves propogating through the elements with possible serious consequences. It is possible to complete relatively simply stress evaluation for suddenly applied and impact loads by using the principle of conservation of energy and conditional that the materials considered are operating within their elastic regions. [www.roymech.co.uk/Useful_Tables/Fatigue/Mechanics_Impact.html].

Most real world impacts are biaxial rather than unidirectional. Further complication is offered by the choice of failure modes: ductile or brittle. Brittle materials take little energy to start a crack, little more to propagate it to a shattering climax. Other materials possess ductility to varying degrees. Highly ductile materials fail by puncture in drop weight testing and require a high energy load to initiate and propagate the crack.

Many materials are capable of either ductile or brittle failure, depending upon the type of test and rate and temperature conditions. They possess a ductile/brittle transition that actually shifts according to these variables. [web@instron.com]

2.5 FINITE ELEMENT METHOD (FEM)

The finite element method or FEM also referred to as finite element analysis is a numerical technique for finding approximate solutions of partial differential equations as well as of integral equations. Or simply, Finite element analysis is a computerized method for predicting how a real world object will react to forces, heat, vibration, and so on, in terms of where it will fails. It is called analysis, but in the product design cycle it is used to predict what is going to happen when the product is used. [Wikipedia/ FEM].

The finite element method works by breaking a real object down into a large number of elements, such as little cubes. The behavior of each little element, which is regular in shape, is readily predicted by set mathematical equations. The computer then adds up all of the individual behaviors to predict the behavior of the actual object. The finite in finite element analysis comes from the idea that there are a finite number of elements in a finite element model. Finite element method is employed to predict the behavior of things with respect to virtually all physical phenomena such as Mechanical stress, Mechanical vibration, Heat transfer, Fluid, Various electrical and magnetic phenomena and Acoustics. [Algor Help File, Algor corp].

The finite element method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the domain. Boundary values problems are also sometimes called field problems. The field variables are the independent variables of interest governed by the differential equation. The boundary condition is the specified values of the field variables or related variables such as derivatives on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only few. [Hutton, David V., 2004]

2.5.1 Nodes and Elements

A node is a coordinate location in space where the degrees of freedom are defined. The degree of freedom represents the possible movement of this point due to the loading of the structure. The DOFs also represent which forces and moments are transferred from one element to the next. The results of a finite element analysis - deflections and stresses, are usually given at the nodes. In the real world, a point can move in 6 different directions, translation in X, Y, and Z, and rotation about X, Y, and Z. In FEA, a node may be limited in the calculated motions for a variety of reasons. For example, there is no need to calculate the out of plane translation on a 2-D element; it would not be a 2-D element if its nodes were allowed to move out of the plane. [Hutton, David V., 2004].

An element is the basic building block of finite element analysis. There are several basic types of elements. An element is a mathematical relation that defines how the degrees of freedom of a node relate to the next. These elements can be lines- trusses or beams, areas- 2D or 3D plates and membranes or solids- bricks or tetrahedrals. It also relates how the deflections create stresses. [Widas, Peter., 1997]

2.5.2 Basic Step in Finite Element Analysis

The basic steps in a finite element analysis are divided into 3 major steps that are setting up the model, analyzing the model and results evaluation. Each step has its own division where for the setting up the model at first the model must be meshed, and then the units are defined. After that the model analysis parameter and the element type parameter are defined. Lastly for this step the load and the boundary conditions are applied. Move on to the second step where the assembly of the stiffness matrix is done followed by solving the linear algebraic equations. Then the results are calculated. For the third step results evaluation is done by reviewing the results and generate a report for the analysis.

2.5.3 Advantages of Finite Element Analysis